Abstract

Convergent-Divergent (C-D) nozzles are used in rocket engines to produce thrust as a reaction to the acceleration of hot combustion chamber gases in the opposite direction. To maximize the engine performance at high altitudes, large area ratio, *bell-shaped* or *contoured* nozzles are used. At lower altitudes, the exit pressure of these nozzles is lower than the ambient pressure. During this over-expanded condition, the nozzle-internal flow adapts to the ambient pressure through an oblique shock. But the boundary layer inside the divergent portion of the nozzle is unable to withstand the pressure rise associated with the shock, and consequently flow separation is induced.

Numerical simulation of separated flows in rocket nozzles is challenging because the existing turbulence models are unable to correctly predict shock-induced flow separation. The present thesis addresses this problem. Axisymmetric, steady-state, Reynolds-Averaged Navier-Stokes (RANS) simulations of a conical nozzle and three sub-scale contoured nozzles were carried out to numerically predict flow separation in over-expanded rocket nozzles at different nozzle pressure ratios (NPR). The conical nozzle is the JPL 45°-15° and the contoured nozzles are the VAC-S1, the DLR-PAR and the VAC-S6-short. The commercial CFD code ANSYS FLUENT 13 was first validated for simulation of separated cold gas flows in the VAC-S1 nozzle. Some modeling issues in the numerical simulations of flow separation in rocket nozzles were determined. It is recognized that compressibility correction, nozzle-lip thickness and upstream-extension of the external domain are the sources of uncertainty, besides turbulence modeling.

In high-speed turbulent flows, compressibility is known to affect dissipation rate of turbulence kinetic energy. As a consequence, a reduction in the spreading rate of supersonic mixing layers occurs. Whereas, the standard turbulence models are developed and calibrated for incompressible flows and hence, do not account for this effect. ANSYS FLUENT uses the compressibility correction proposed by Wilcox [1] which modifies the turbulence dissipation terms based on turbulent Mach number. This, as shown in this thesis, may not be appropriate to the prediction of flow separation in rocket nozzles. Simulation results of the standard SST model, with and without the compressibility correction, are compared with the experimental data at NPR=22 for the DLR-PAR nozzle. Compressibility correction is found to cause under-prediction of separation location and hence its use in the prediction of flow separation is not recommended.

In the literature, computational domains for the simulation of DLR subscale nozzles have thick nozzle-lips whereas for the VAC subscale nozzles they have no nozzle-lip. Effect of nozzle-lip thickness on flow separation is studied in the DLR-PAR nozzle by varying its nozzle-lip thickness. It is found that nozzle-lip thickness significantly influences both separation location and post-separation pressure recovery by means of the recirculation bubbles formed at the nozzle-lip.

Usually, experimental values of free stream turbulence are unknown. So conventionally, to minimize solution dependence on the boundary conditions specified for the ambient flow, the computational domain external to the nozzle is extended in the upstream direction. Its effect on flow separation is studied in the DLR-PAR nozzle through simulations conducted with and without this domain extension. No considerable effect on separation location and pressure recovery is found.

The two eddy-viscosity based turbulence models, Spalart-Allmaras (SA) model and Shear Stress Transport (SST) model, are well known to predict separation location better than other eddy-viscosity models, but with moderate success. Their performances, in terms of predicting separation location and post-separation wall pressure distribution, were compared with each other and evaluated against experimental data for the conical and two contoured nozzles. It is found that they fail to predict the separation location correctly, exhibiting sensitivity to the range of NPRs and to the type of nozzle.

Depending on NPR, the SST model either under-predicts or over-predicts Free Shock Separation (FSS). Moreover, it also fails to capture Restricted Shock Separation (RSS). With compressibility correction, it under-predicts separation at all NPRs to a greater extent. Even though RSS is captured by using compressibility correction, the transition from FSS to RSS is over-predicted [2]. Early efforts by few researchers to improve predictions of nozzle flow separation by realizability corrections to turbulence models have not been successful, especially in terms of capturing both the separation types.

Therefore, causes of turbulence modeling failure in predicting nozzle flow separation correctly were further investigated. It is learnt that limiting of the shear stress inside boundary layer, due to Bradshaw's assumption, and over-prediction of jet spreading rate are the causes of SST model's failure in predicting nozzle flow separation correctly. Based on this physical reasoning, values of the a_1 parameter and the two diffusion coefficients $\sigma_{k,2}$ and $\sigma_{\omega,2}$ were empirically modified to match the predicted wall pressure distributions with experimental data of the DLR-PAR and the VAC-S6-short nozzles. The results confirm that accurate prediction of flow separation in rocket nozzles indeed depends on the correct prediction of spreading rate of the supersonic separation-jet. It is demonstrated that accurate RANS simulation of flow separation in rocket nozzles over a wide range of NPRs is feasible by modified values of the diffusion coefficients in turbulence model.